Defence Science Journal, Vol. 67, No. 2, March 2017, pp. 219-223, DOI : 10.14429/dsj.67.9544

Numerical Characterisation of Supersonic Exhaust Diffusers

M. Srinivasa Rao, Afroz Javed*, and Debasis Chakraborty

Directorate of Computational Dynamics, Defence Research and Development Laboratory, Hyderabad - 500 058, India
**E-mail: afrozjaved@gmail.com*

ABSTRACT

Rocket motors that are designed to operate at high altitudes need a nozzle with a large expansion ratio to maximize the thrust at much lower atmospheric pressure than that of the sea level pressure. Accurate performance of these nozzles cannot be obtained when static tested on ground. Computational fluid dynamics (CFD) analyses have been performed to characterise the supersonic exhaust diffuser (SED). The results obtained from the CFD analyses have been found to be in good agreement with experimental and numerical values reported in the published literature. Started and un-started regions of the SED have been identified with the CFD results.

Keywords: Computational fluid dynamics; High altitude; Supersonic exhaust diffuser; Rocket nozzles

1. Introduction

Rocket nozzles designed for high-altitude operation have high throat to exit area ratio for maximum utilisation of the chamber pressure. When these rocket motors are tested in ground level conditions the exhaust gas flow separates in the divergent portion of the nozzle. The criterion for this separation was first proposed by Summerfield[1], *et al.* and it remains in use for first estimation of flow separation. A review of several other semi-empirical criteria proposed by different researchers for the prediction of flow separation is collated by Stark[2]. All of these criteria are found to be of limited use and only a qualitative prediction of flow separation can be made, making it difficult to predict the exact location and resulting ground level performance of the rocket nozzles. Numerical prediction of the flow separation has also been attempted[3-6] over the years using RANS methodology and it is observed that the separation point location is very sensitive to the choice of turbulence model and the model constants, making the validity of predictions for general cases not to be accurate. The other methods like LES and DNS are not used because of very high velocity flows and consequent requirement of large grid and computational cost[7].

Due to these limitations on the prediction capability of available numerical tools, it becomes desirable to carry out static tests simulating high altitude pressure conditions at the exit of the nozzle. One method to simulate high altitude test conditions is a supersonic exhaust diffuser (SED) and another is an ejector system. The simpler method of these is using an SED. An SED consists of a vacuum chamber, cylindrical supersonic diffuser and a subsonic diffuser. The working principle of the SED is explained in detail in literature[8-10]. A schematic layout of an SED with a rocket motor to be tested is as shown in Fig. 1. The impingement of the rocket jet on SED wall and vacuum level in the chamber is shown to be a function of nozzle throat area (A_t), duct cross sectional area (A_d), duct exit area (A_e), and ratio of the specific heats (γ) of the products of combustion[9].

At lower values of chamber pressure when the jet momentum exhausted from rocket is not enough, the flow is separated from the nozzle wall. With increase in chamber pressure, the nozzle flows full but over-expanded, however, the diffuser still remains unstarted. The unstarted regime consists of two phases. In the first phase, the flow separates from the nozzle walls through oblique shock, and in the second phase, the flow separation is at the nozzle exit. As the chamber pressure is further increased, the diffuser also flows full so that the shock system is fully established in the duct. In this regime, the supersonic jet from the nozzle impinges on the diffuser wall separating the effect of ambient pressure from the vacuum chamber through a system of shocks. At this stage, the supersonic exhaust diffuser is said to have started and the corresponding chamber pressure is the minimum starting pressure. Due to ejector effect, gas is sucked from upstream direction of the nozzle and a vacuum is created in the vacuum chamber. The vacuum level in the vacuum chamber goes on increasing with the rocket chamber pressure, but as soon as the diffuser is started there is no change observed in the vacuum level and it remains constant.

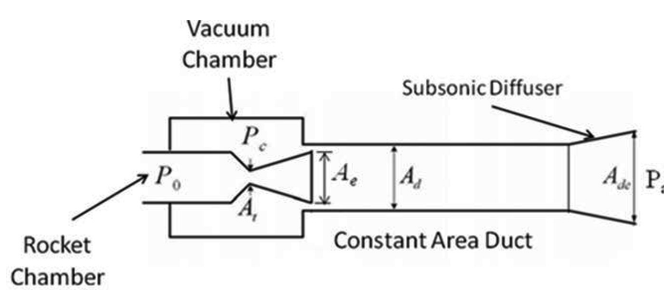

Figure 1. Schematic sketch of supersonic exhaust diffuser.

Received : 20 December 2015, Revised : 16 November 2016
Accepted : 13 February 2017, Online published : 11 March 2017

The size of the SED can be designed using one dimensional steady state adiabatic assumptions without considering frictional losses. SED performance with the absence of these simplifying assumptions can be evaluated using numerical simulations for the flow field. In the present work two different SED geometries are studied using CFD methodology.

2. GEOMETRIES AND GRIDS

There are two different geometries considered for the numerical study of SEDs[9-10]. Both these geometries conform to the schematic sketch as shown in Fig. 1, except that the geometry described in[10] does not have subsonic diffuser part and contains only a straight duct. The geometric parameters of both these cases are listed in Table 1. The throat diameter of the rocket nozzle is represented by D_t, the diameter of constant area duct is represented by D, while L is the length of the diffuser duct measured from nozzle exit.

Since the supersonic exhaust diffuser is a cylindrical duct, the domain of analyses has been taken to be axi-symmetric. Grid has been generated using ANSYS ICEM CFD 14.5[11]. The grid contains hexahedral cells with clustering near the wall. Three different grid sizes of 16000, 34000, and 78000 have been studied. The details of CFD methodology for these simulations are in section 3.0. The simulations are carried out for an operating pressure of 50 bar. The wall pressures are plotted in Fig. 2. It can be seen that the differences in the wall pressures for 34000 and 78000 grid sizes are very small. Hence the grid size of 34000 is chosen for the further simulations. The y^+ obtained for this grid size is less than 1. These values of y^+ ensure a sufficiently fine grid for this class of problems.

3. CFD METHODOLOGY

Numerical analyses have been carried out using ANSYS Fluent 14.5[12] software. This software solves 3-dimensional Reynolds Averaged Navier Stokes (RANS) equations using finite volume methodology. Both pressure based and density based solvers can be used. Density based solvers are recommended for compressible flows[13]. As most of the flow regime in the present problem is in supersonic condition, density based solver has been used for the analyses. Second order upwind scheme is used for discretising the convection terms while diffusion terms are discretised using second order central differencing scheme, for the governing transport equations. Since the governing equations are nonlinear and coupled to one another, the solution process involves iterations wherein the entire set of governing equations is solved repeatedly until a converged solution is obtained. The continuity, momentum, and energy equations are solved simultaneously (in a coupled manner), while turbulence equations are solved using the previously updated values of the other variables. After solving these equations a check is made for convergence of the variables, in case of non convergence the process is repeated with updated values of the variables. In the present simulations the convergence criterion is taken as 10^{-6} for all the equations.

The averaging process of Navier Stokes equations for a single species gives rise to additional terms in momentum equation, called Reynolds stresses. The additional term in the energy equation is called Reynolds heat flux vector. These additional terms require modelling. The Reynolds stress term is modelled through the concept of eddy viscosity. The value of eddy viscosity is evaluated by using a two equation turbulence transport model. In the present case k-ε turbulence model suggested by Launder and Spalding[14] is used to evaluate eddy viscosity, due to its inherent robustness and stability. While the k-ε turbulence model provides good prediction for many flows of engineering interest, it is not suitable for applications with boundary layer separation, rotating fluids, flow on curved surfaces, flows with sudden change in mean strain rates, and free shear flows with compressible convective Mach numbers[14,15]. The Reynolds heat flux vector is modelled through the concept of Turbulent Prandtl number. The value of turbulent Prandtl number in the present simulations is taken as 0.9 as the flow is dominated by the interaction with solid boundaries of nozzle and diffuser walls[16].

High pressure nitrogen gas at normal temperature is utilised in both the experimental cases[9-10] being considered in the present study. The supply pressure of the gas is taken as chamber pressure of the rocket motor (P_0 in Fig. 1), and is applied as total pressure inlet boundary condition. The inlet total temperature is taken as 300 K. Outlet boundary is given as pressure boundary with atmospheric pressure (P_a in Fig. 1). The vacuum chamber pressure (P_c in Fig. 1) is recorded after convergence of each run.

Table 1. Geometric details of the supersonic exhaust diffuser

Case	D_t (mm)	D (SED diameter) (mm)	A_e/A_t	A_d/A_e	A_d/A_t	L/D
Geometry 1[9]	2.80	21.00	35.02	1.61	56.25	12.38
Geometry 2[10]	8.87	52.20	20.97	1.65	34.60	8.00

4. RESULTS AND DISCUSSIONS

Mach number variations inside SED along with the formation of the jet from the nozzle at various conditions of P_0/P_a have been shown in Fig. 3. Analyses have been performed for P_0/P_a = 10 to 60. Shock structure is found to be more

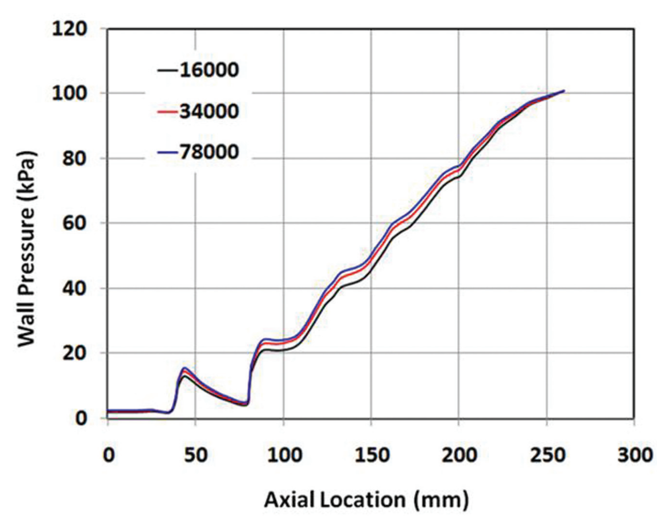

Figure 2. Surface pressures on the constant area duct wall for different grid sizes.

complex with increase in P_0/P_a. In the region $P_0/P_a < 40$, both nozzle and diffuser are found to be in un-started conditions and the recirculation was dominant at the exit of the nozzle. At $P_0/P_a = 40$, the nozzle and diffuser were found to be in started condition and a series of complex shock structures have been observed. In this zone, oblique shocks impinge on the diffuser walls and isolate the vacuum chamber from the atmospheric conditions and further increase in P_0/P_a does not cause any appreciable change in the vacuum chamber pressure.

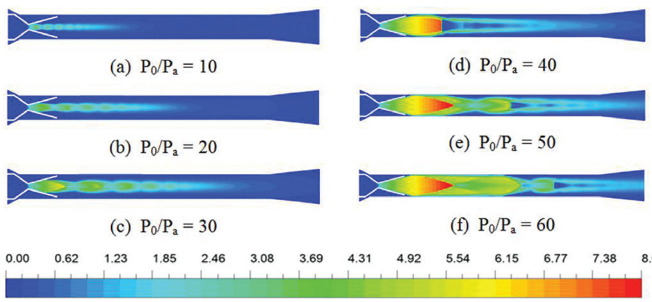

Figure 3. Contours of Mach number along the length of the SED.

The wall pressures obtained from the CFD analyses is compared with the experimental and CFD computations[9] and shown in Fig. 4. The CFD computations[9] are made using an in house code which solves RANS equations along with *k-ε* turbulence. The *k-ε* turbulence model used[9] is based on the formulation proposed by Yang and Shih[17]. The simulations have used dual time stepping to address issues involved with wide range of velocity scales, i.e., supersonic speeds in SED duct and near stagnation in the vacuum chamber. Notwithstanding these minor differences, it can be observed from Fig. 4, that the results of present CFD analyses are in good agreement with published results[9]. First impingement of the oblique shocks from the lips of the nozzle walls over the diffuser walls have been identified as *X* in Fig. 4. The deviation of the predicted results from that of experimental values is found to be less than 2 per cent.

Characteristics of SED have been well predicted by the simulations and the results are compared with experimental values and the CFD results[9] in Fig. 5. The pressures in the vacuum chamber are recorded for all the P_0/P_a values. At lower P_0/P_a, the jet flow through the nozzle is un-started and so is the diffuser. The un-started zone has been identified from A-B in Figure 4, P_0/P_a varies from 10 to 40, and started region is identified as B-C, and it ranges from P_0/P_a varies from 40 to 60.

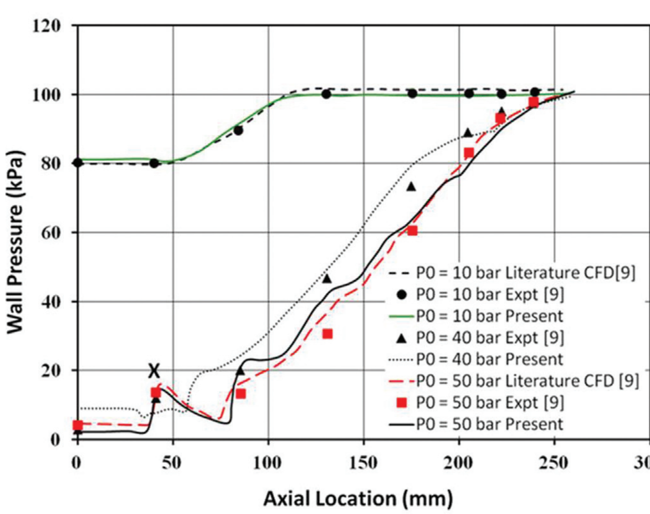

Figure 4. Comparison of wall pressures for different pressure ratios.

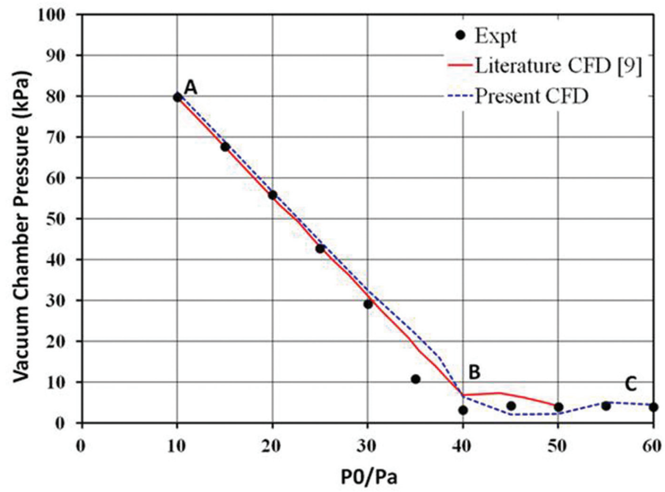

Figure 5. CFD results and experimental data for the characteristics of SED in first case.

Although there are differences near and after the starting point of the diffuser, computations capture the trend of the vacuum chamber pressures well for different pressure ratios. Also, experimental diffuser wall pressures in the started region (P_0 = 40 and 50 bar) corroborate well with the computed results (Fig. 4). The starting process of the diffuser is a complex phenomena and its prediction requires time accurate simulation involving very small time step which has not been attempted in the present study.

The second geometry[10] considered for numerical analysis has a larger throat diameter of the nozzle with a smaller expansion ratio as compared with the first case geometry. The size of the duct is also larger. The ratios of nozzle exit areas to SED duct cross section areas are nearly same for the two geometries (differing by nearly 2.5 per cent). CFD simulations have been carried out for this configuration also and the results are compared with the experimental data. Fig. 6, shows the comparison of the CFD and experimental results for the variation of P_c/P_a with respect to variation in P_0/P_a.

It can be seen in Fig. 6 that as the motor chamber pressure increases the differences between the vacuum chamber pressures obtained experimentally and those obtained from numerical simulations go on increasing till the starting of diffuser. The starting of the diffuser from experimental results occurs at P_0/P_a = 29.20, however the same is predicted to be at P_0/P_a = 27.07 from CFD simulations differing by around 7 per cent. The difference is conjectured to be due to a small difference of (A_e/A_t) in the computational and experimental models. No attempts have been made to reduce the difference in the present study.

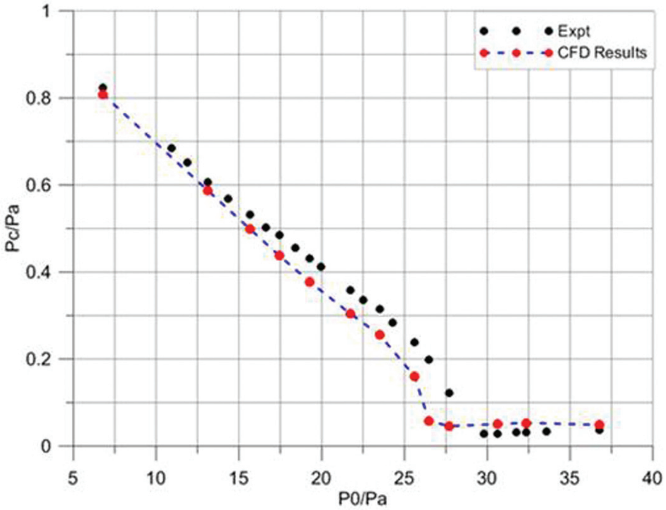

Figure 6. CFD results and experimental data for the characteristics of SED in second case.

5. CONCLUSIONS

CFD analyses have been performed to characterise the supersonic exhaust diffuser (SED). Two model geometries reported in literature are considered for the numerical study. A number of numerical simulations with different values of rocket chamber pressures. A density based solver is used with second order discretisation and *k-ε* turbulence model for the numerical studies. For the first case it has been observed that the wall pressures predicted by CFD methodology match well within 2 per cent of their values with those measured experimentally for different values of chamber pressure. Also a good match is observed for the prediction of the starting of diffuser with chamber pressure. In the second case the diffuser starting pressure predicted by CFD methodology is 7 per cent less than that evaluated experimentally. With the consideration of experimental uncertainties in the experimental values and numerical accuracies of the CFD methodology the predictions can be considered good and CFD can be used for the design and characterisation of the SEDs.

REFERENCES

1. Summerfield, M.; Foster, C.; & Swan, W. Flow separation in over expanded supersonic exhaust nozzles. *Jet Propulsion*, 1954, **24**(9), 319-321.
2. Ralf, Stark. Flow separation in rocket nozzles – An Overview. 2013, AIAA Paper No. 2013-3840.
doi: 10.2514/6.2013-3840.
3. Hadjadj, A.; Nebbache, A.; Vuillamy, D. & Vandromme, D. Numerical simulation of flow separation in rocket nozzle. *Mech. Res. Commun.*, 1997, **24**(3), 269-276.
doi: 10.1016/s0093-6413(97)00024-4.
4. Nasuti, F.; Onofri, M. & Martelli, E. Numerical analysis of flow separation structures in rocket nozzles. 2007, AIAA Paper No. 2007-5473. doi: 10.2514/6.2007-5473.
5. Allamaprabhu, C.Y. & Raghunandan, B.N. Improved prediction of flow separation in thrust optimized parabolic nozzles with FLUENT. 2011, AIAA paper No.2011-5689,
doi: 10.2514/6.2011-5689.
6. Xiao, Q.; Tsai, H.M. & Papamoschou, D. Numerical investigation of supersonic nozzle flow separation. *AIAA Journal*, 2007, **45**(3), 532-541.
doi: 10.2514/1.20073.
7. Ostlund, J. & Muhammad-Klingmann, B. Supersonic flow separation with application to rocket engine nozzles. *Applied Mech. Rev.*, 2005, **58**, 143-177.
doi: 10.1115/1.1894402.
8. Roschke, E.J.; Massier, P.F. & Geer, H.L. Experimental investigation of diffusers for rocket engines. JPL Report TR 32-210, 1962.
9. Sung, H.; Yoon, S.; Yeom, H.; Jinkon, ChungNam, Kim, Y. & Seunghyup. Study on design - and operation - Parameters of supersonic exhaust diffusers. 2008, AIAA paper No. 2008-855.
doi: 10.2514/6.2008-855.
10. Annamalai, K. ; Visvanathan, K. ; Sriramulu, V.& Bhaskaran, K.A. Evaluation of the performance of supersonic exhaust diffuser using scaled down models. *Experimental Thermal and Fluid Science*, 1998, **17**, 217-229.
doi: 10.106/s0894-1777(98)00002-8.
11. User manual ANSYS ICEM CFD 14.5.
12. User manual ANSYS Fluent 14.5.
13. Merkle, C.L.; Venkateswaran, S. & Buelow, P.E.O. The relationship between pressure – based and density – based algorithms. 1992, AIAA paper No. 92-0425,
doi: 10.2514/6.1992-425.
14. Launder, B.E. & Spalding, D.B. Lectures in mathematical models of turbulence. Academic Press, London, England. 1972.
15. Launder, B.E.; Morse, A.; Rodi, W. & Spalding, D.B. Prediction of free shear flows – a comparison of the performance of six turbulence models. *In* Proceedings of NASA Conference on *Free Shear Flows* 1972; Langley AFB.
16. Wilcox, D.C. Turbulence modeling for CFD, DCW Industries Inc, La Canada, 1994, p 182.
17. Yang, Z. & Shih, T.H. New time scale based *k* −ε model for near-wall turbulence. *AIAA Journal*, 1993, **31**(7) 1191-1197.
doi: 10.2514/3.11752

CONTRIBUTORS

Mr M. Srinivasa Rao did his BTech from J.N.T. University, Hyderabad, in 2002 and MTech from National Institute of Technology (NIT) Bhopal, in 2004. Presently he is working in the Directorate of Ramjet Propulsion, Defence Research & Development Laboratory (DRDL), Hyderabad. He has published 13 conference papers. His research interests include: Computational fluid dynamics (CFD) and propulsion.
In the present study, he has contributed in generating the grid, performed numerical simulations and post processed the results.

Dr Afroz Javed did his MTech from Indian Institute of Technology (IIT) Madras, in 1996 and PhD from Indian Institute of Science (IISc), Bengaluru, in 2013. Presently he is working in the Directorate of Computational Dynamics, Defence Research & Development laboratory (DRDL), Hyderabad. His research interests include: Computational fluid dynamics (CFD), propulsion, combustion instability, and turbulent compressible mixing.

In the current study, he participated in the simulation, results analysis and manuscript preparation.

Dr Debasis Chakraborty obtained his PhD (Aerospace Engineering) from Indian Institute of Science (IISc), Bengaluru. Presently, he is working as Technology Director, Computational Fluid Dynamics (CFD), DRDl, Hyderabad. His research areas includes: CFD, aerodynamics, high speed combustion and propulsion. He has more than 215 technical papers in referred journals and conferences.

In the current study, he was involved in the problem formulation, analysis of the results and finalisation of the manuscript.